

SolidWorks Flow Simulation: Selecting the optimal mesh for conducting CFD analysis on a centrifugal fan

Salmat*, Delima yanti Sari, Yolli Fernanda and Febri Prasetya

Department of Mechanical Engineering, Faculty of Engineering, Universitas Negeri Padang,
INDONESIA

Abstract: The accuracy of mesh use determines the suitability of simulation results with experimental test results, therefore it is necessary to study the type and number of mesh used for flow simulation. This study aims to investigate the type of mesh and the number of cells that are appropriately used in the simulation of centrifugal fans using the Solidworks Flow Simulation module. The research was conducted by comparing the simulation results with the experimental results on centrifugal fans that have been carried out by previous studies. In this research, the type of mesh used is global mesh. The parameter of determining the number of cells is done by varying the mesh level because the mesh level affects the number of cells. In this simulation study, the boundary condition is set to one flow volume of 5 m³/h and a rotation speed of 92.362 rad/s. In this study, the total pressure and efficiency of the centrifugal fan are the things that are seen from the comparison of flow simulation with the experiment. Based on the simulation results that have been carried out, mesh levels 1 to 4 show results that differ significantly from the experimental results. At machine levels 5 to 7, the results are close to the experimental results, but the closest total pressure and efficiency values are at mesh level 5. The results of this study can be a reference for research that simulates centrifugal fans.

Keywords: Mesh validation; Convergence; Pressure; Efficiency

*Corresponding Author: salmat1710@student.unp.ac.id

Received: June 26th 2023; Revised: September 17th 2023; Accepted: October 02nd, 2023

<https://doi.org/10.58712/jerel.v2i3.104>

Reference to this paper should be made as follows: Salmat, S., Yanti Sari, D., Fernanda, Y., & Prasetya, F. (2023). Optimization of impeller blade number in centrifugal pump for crude oil using Solidworks Flow Simulation SolidWorks Flow Simulation: Selecting the optimal mesh for conducting CFD analysis on a centrifugal fan. *Journal of Engineering Researcher and Lecturer*, 2(3), 94-103. <https://doi.org/10.58712/jerel.v2i3.104>

1. Introduction

Centrifugal fans are fluid delivery devices that have been widely used in various engineering fields ([Singh et al., 2011](#)). Since they were first invented, centrifugal fans have been the focus of development for many academics and engineers to date. The working principle of a centrifugal fan involves fluid flow entering the inlet axially towards the impeller, and then exiting radially ([Cory, 2005](#)). These devices have an important role as one of the key components in the HVAC industry ([Patel et al., 2014](#)). Early in their development, centrifugal fan designs were often based on the assumption of one- or two-dimensional idealized flow ([Eck, 1952](#)). However, in reality, there are complex three-dimensional flows in centrifugal fans, making the previously used methods inefficient in terms of design time and cost ([Yu et al., 2005](#)). In particular, for fans used in HVAC systems, the pressure increase at the outlet tends to be small, so the numerical method used in modelling must be accurate enough to capture the small pressure increase. Nowadays, technological developments have brought three-dimensional fluid flow prediction or Computational Fluid Dynamics (CFD) to be a reliable method for analyzing the performance of centrifugal fans. The use of CFD to solve three-dimensional flow calculation problems in centrifugal fans has proven to be effective and efficient.

Nowadays, many scholars have used CFD to solve centrifugal fan performance problems to obtain an optimal design. The standard k-ε calculation model is used in various numerical

simulation techniques of fluid flow ([Lee et al., 2011](#); [Lin & Huang, 2002](#); [Yu et al., 2005](#)). Several studies have validated the effectiveness of the k- ϵ calculation model in predicting centrifugal fan performance, such as that of ([Seo & Kim, 2009](#)), who successfully applied the k- ϵ model in CFD simulations for a multi-blade type centrifugal fan and revealed increases in total pressure, shaft power, and efficiency. ([Yu et al., 2005](#)) applied the k- ϵ model to reveal the effect of blade angle and impeller gap. In another study, ([Huang & Hsieh, 2009](#)) successfully designed the optimal design of a backward-curved airfoil impeller using the k- ϵ model in CFD simulations. ([Thakur et al., 2002](#)) used CFD simulation to analyze the fluid flow in a centrifugal blower to determine the flow instability of the rotating impeller. Based on the study data, the researchers concluded that the fluid flow prediction is generally accepted. Many CFD tools help use the k- ϵ model to predict three-dimensional flow in centrifugal fans, and one of the commonly used packages in many studies is ANSYS ([Huang & Hsieh, 2009](#); [Kulkarni et al., 2016](#); F. [Meng et al., 2013](#); F.-N. [Meng et al., 2017](#); [Yu et al., 2005](#)). Despite ANSYS being the most commonly used tool in flow prediction in centrifugal fans, many other CFD tools can be used, such as Solidworks Flow Simulation.

Solidworks Flow Simulation uses the Navier-Stokes equation to predict fluid flow ([Sobachkin, 2014](#)). Solidworks Flow Simulation can be used to accurately predict fluid flow on airfoils ([Wallace, 2019](#)). It can be said that currently alternatively Solidworks Flow Simulation can be used to predict fluid flow in centrifugal fans. in this study aims to prove. However, there are not many uses of Solidworks Flow Simulation to predict the flow in centrifugal fans.

This study aims to obtain the right mesh type and number of cells that should be used when performing CFD on a centrifugal fan at the Solidworks Flow Simulation. This research is conducted by validating simulation results with experimental results. One of the methods used to check the accuracy of the mesh and the number of cells used is the independent mesh test. However, in this method, simulation results are not compared with experimental results, so the agreement between simulation results and experimental results is not visible. The independent mesh test aims to show that the mesh count does not affect the simulation results, as well as to consider the time used in the simulation ([Kulkarni et al., 2016](#)). To validate the accuracy of CFD simulation results, experimental test results are required ([Huang & Hsieh, 2009](#); [Kulkarni et al., 2016](#); [Lee et al., 2011](#); [Lin & Huang, 2002](#); [Nabawi et al., 2021](#); [Oberkampf & Trucano, 2002](#); [Odyjas & Moczko, 2022](#); [Yu et al., 2005](#)). In this study, the centrifugal fan design used to compare simulation results with experimental results is based on research conducted by Huang & Hsieh (2009).

2. Material and methods

In this study, the CFD software utilized was Solidworks Research License 2021-2022. The research process began with the modelling of a centrifugal ventilator, which was based on an experimental study published by ([Huang & Hsieh \(2009\)](#)). Validation was done by comparing the simulation results with the recorded experimental data. The parameters observed in this study include the total pressure and efficiency of the centrifugal ventilator.

2.1 Design of Centrifugal fan

The model used in this study refers to a previous study by ([Huang & Hsieh in 2009](#)). The three-dimensional model was created using a computer-aided design (CAD) tool in Solidworks, with the model parameters listed in Table 1 and shown in Fig. 1. In the inlet section, a length of 10 times the diameter was set, while in the outlet section, the length was set at 15 times the diameter. It is assumed that this will result in stable fluid flow entering and exiting the fan ([Huang & Hsieh, 2009](#)). The impeller in this study rotates with a constant rotation speed, so it is defined as a rotation region with a sliding rotation type.

Table 1. Main parameters of centrifugal fan

Parameters	Value
Diameter Luar Impeller, ^a mm	910
Blade angle, ^b degrees	46.39
Blade Number	12
Tongue Length, ^c mm	214
Impeller width, ^d mm	224
Inlet Diameter, ^e mm	924
Outler Size, ^f mm x mm	824 x 737

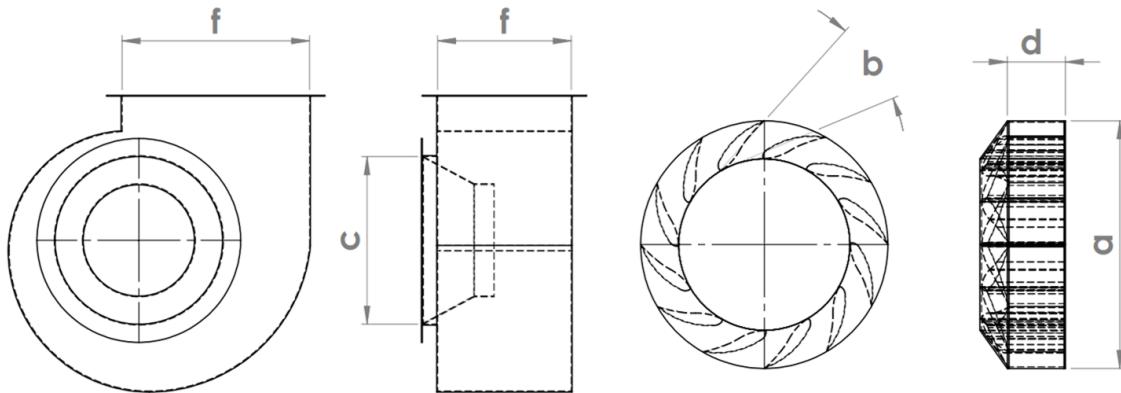


Figure 1. Centrifugal fan design (Huang & Hsieh, 2009)

2.2 Mesh

This study established seven mesh levels, where each increase in mesh level resulted in a higher number of cells. Mesh level in this SolidWorks Flow Simulation is the process of dividing the analysed component into smaller cells. The resulting cell counts of the seven variations are outlined in Table 2, and the application of the mesh results is shown in Fig. 2.

Table 2. The number of meshes at each level

Level	1	2	3	4	5	6	7
Number of Cells	1957	16019	24188	76313	214569	551188	1393953

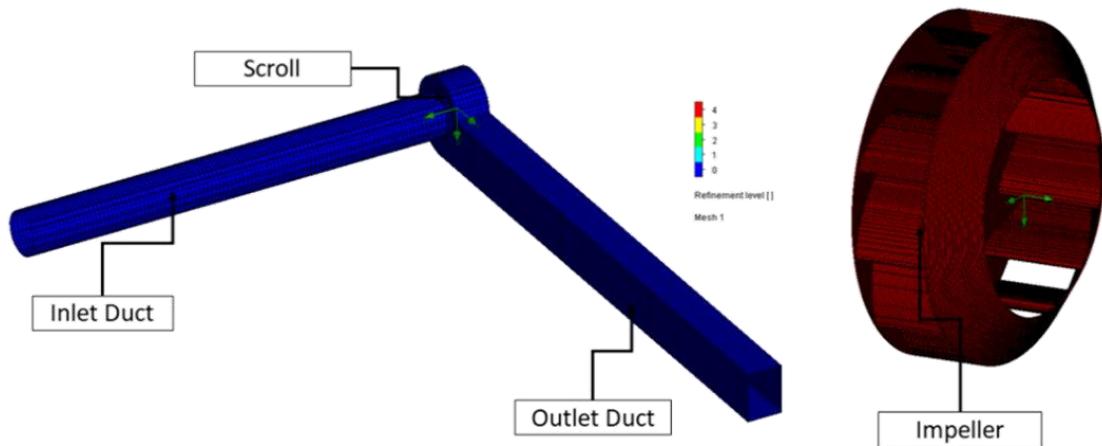


Figure 2: Meshing

2.3 Boundary condition

This study uses several parameters that refer to the conditions implemented in the experiments described by ([Huang & Hsieh, 2009](#)). The flow parameters used in this study include incompressible flow, no-slip boundary conditions, neglect of gravity effects, and neglect of fluid temperature. In this model, the boundary conditions applied include giving a volume flow rate (Q) at the inlet, which ranges from rest to up to 5 m³/h, while at the outlet, the total pressure (ΔP) is kept at the same level as atmospheric pressure. In addition, the impeller is set to a sliding condition in the local region, with an angular velocity (ω) of 92.362 rad/s. An illustration of the boundary condition setting in the simulation model can be found in Fig. 3.

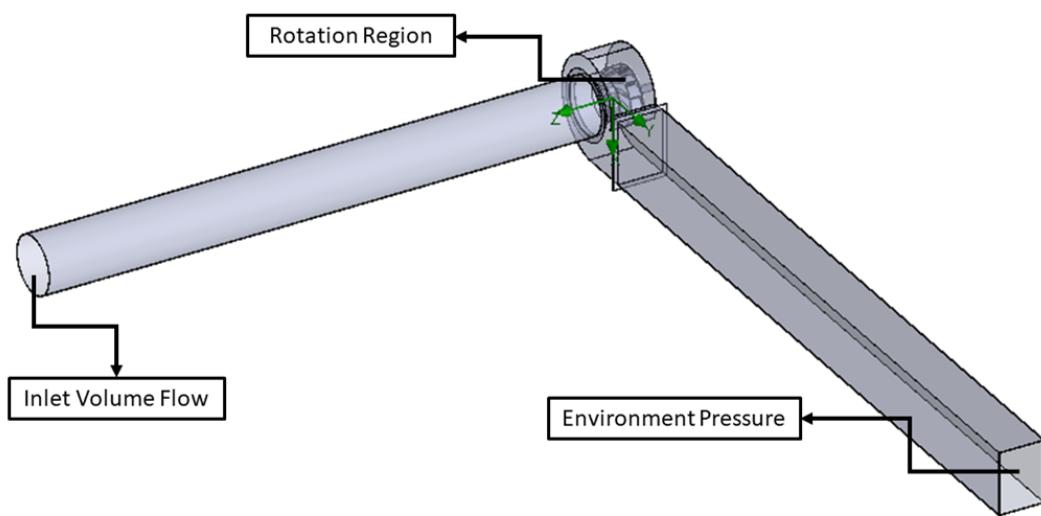


Figure 3. Boundary condition

2.4 Fundamental equations

The performance of a centrifugal fan can be measured through various parameters, including total pressure rise, shaft power, and efficiency at various airflow rates. The total pressure rise is defined as the difference between the average pressure at the fan inlet and outlet. In this study, the total efficiency of the fan is described as described in [Huang & Hsieh, 2009](#) study, as seen in Eq. 1.

$$\eta = \frac{P_0 \times Q}{W \times 3600} \quad (1)$$

The shaft power supplied to the fan can also be expressed in brake horsepower (bhp), which is defined by Eq. 2.

$$bhp = W = \omega \times T_{shaft} \quad (2)$$

2.5 Navier-Sotke equations

Flow simulation in SolidWorks Flow Simulation uses transport equations for turbulent kinetic energy and dissipation rate with a k- ϵ model. To predict the flow conditions, the Navier-Stokes equations are used. The modified k- ϵ model with the damping function described by [Lam & Bremhorst \(1981\)](#) is a mathematical framework that characterises fluid flow behaviour. This model applies the principles of turbulence conservation in fluid dynamics. Its main capability lies in its accuracy in capturing the complex interactions between turbulence and fluid motion. The mathematical description of the Navier-Stokes equations. See Eq. 3-6.

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho k u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right) + \tau_{ij}^R \frac{\partial u_i}{\partial x_j} - \rho \varepsilon + \mu_t P_B, \quad (3)$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial \rho \varepsilon u_i}{\partial x_i} \equiv \frac{\partial}{\partial x_i} \left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right) + C_{\varepsilon 1} \frac{\varepsilon}{k} \left(f_1 \tau_{ij}^R \frac{\partial u_i}{\partial x_j} + C_B \mu_t P_B \right) - f_2 C_{\varepsilon 2} \frac{\rho \varepsilon^2}{k} \quad (4)$$

$$\tau_{ij} = \mu s_{ij}, \tau_{ij}^R = \mu_t s_{ij} - \frac{2}{3} \rho k \delta_{ij}, s_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k}, \quad (5)$$

$$P_B = \frac{g_i}{\sigma_B} \frac{1}{\rho} \frac{\partial \rho}{\partial x_i}, \quad (6)$$

Where, $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1$, $\sigma_\varepsilon = 1.3$, $\sigma_B = 0.9$, $C_B = 1$ if $P_B > 0$

Turbulent viscosity is defined from Eq. 7.

$$\mu_t = f_\mu \times \frac{C_\mu \rho k^2}{\varepsilon} \quad (7)$$

[Lam & Bremhorst \(1981\)](#) defined the dumping function from Eq. 8.

$$f_\mu = (1 - e^{-0.025 R_y})^2 x \left(1 + \frac{20.5}{R_t} \right) \quad (8)$$

Where,

$$R_y = \frac{\rho \sqrt{k y}}{\mu}, \quad (9)$$

$$R_t = \frac{\rho k^2}{\mu \varepsilon} \quad (10)$$

3. Results and discussion

3.1 Convergence on each mesh variation

After performing CFD simulations on each mesh variation, we verified the data by checking the convergence rate. In a study conducted by [Nabawi et al., \(2021\)](#) using Solidworks Flow Simulation, the analysis results were categorised as accurate and valid when the calculation process was repeated until the calculation results were close to zero (normalisation scale 0-1).



Figure 4. Convergece

In Solidworks, when the calculation in the simulation process reaches convergence, it will be marked with "Achieved (IT=...)" in the goal set used. By analysing the results of the CFD simulations performed on each mesh variation, we found that all mesh levels used were able to achieve convergence. The exemplification of the achieved convergence is presented in Fig. 4.

3.2 Discussion Validate simulation results with experimental results

3.2.1 Total of pressure

After conducting the CFD simulation, the results of each variation in mesh level were compared against the experimental findings to assess the concordance between the CFD simulation results and the experimental data. Based on the experimental research of [Huang & Hsieh, \(2009\)](#), the centrifugal fan model with boundary conditions used in the research has also been described above, the experimental results show a total pressure of 700 Pa and an efficiency of 64%, and are shown in Fig. 5 and 6. The results of the research are used to validate the CFD simulation results in this study. Fig. 5 portrays the disparity in total pressure between the simulation results at various mesh levels and the experimental outcomes. Notably, at levels 1 and 2, a substantial deviation from the experimental results is observed, amounting to 85% at level 1 and 51.2% at level 2. Conversely, at levels 3 and 4, the disparities are relatively narrower, standing at approximately 28.52% for level 3 and 35.6% for level 4, respectively. In contrast, for levels 5, 6, and 7, the simulation outcomes exhibit a closer resemblance to the experimental data, with disparities of approximately 16% at level 5, 17.3% at level 6, and 18.86% at level 7. Among the seven mesh level configurations tested, it is advisable to opt for level 5 for future CFD simulations of centrifugal fans.

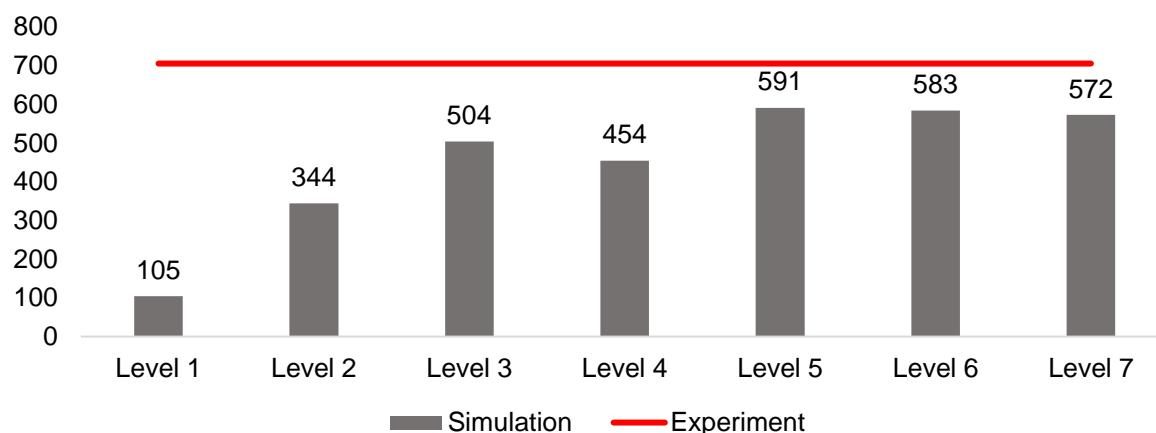


Figure 5. Differences in total pressure at each mesh level with experimental results

3.2.2 Efficiency

Figure 6 show the simulated centrifugal fan efficiency at various mesh levels compared to the experimental results. At levels 1 and 2, there is a significant difference from the experimental results, which is 81% at level 1 and 37.98% at level 2. At levels 3 and 4, there is a closer difference, about 16.18% at level 3 and 19.65% at level 4, respectively. While at levels 5, 6, and 7, the simulation results are close to the experimental results, with differences of about 0.125% at level 5, 1.1% at level 6, and 3% at level 7. It can be concluded that the mesh level setting plays a crucial role in CFD simulation results that match the experimental results. Of the seven mesh level settings tested, level 5 is the recommended one to be used in future CFD simulations of centrifugal fans. However, it should be noted that the higher the mesh level, the longer the estimated time required for simulation tends to be. A higher mesh level also does not guarantee that the accuracy of the simulation results is close to the experimental results.

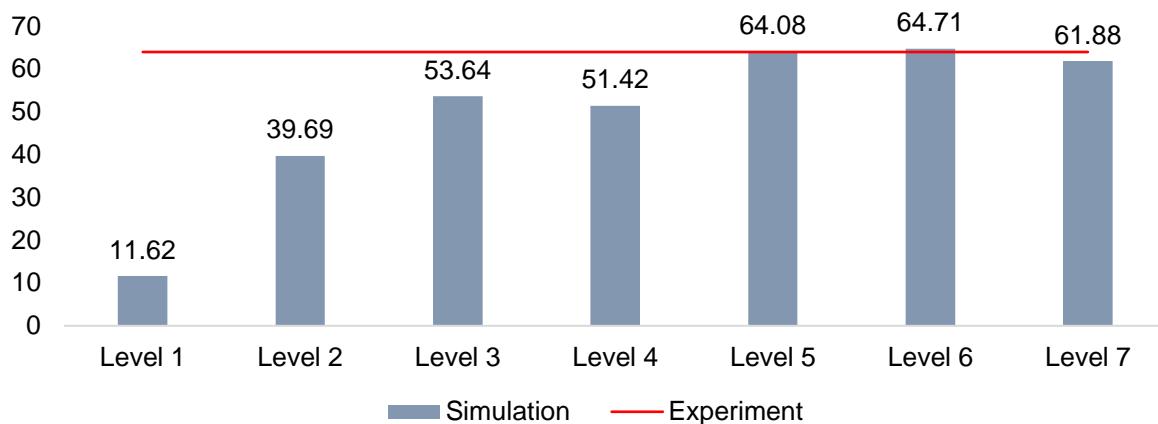


Figure 6. Differences in total efficiency at each mesh level with experimental results

3.3 Flow conditions at mesh level settings

The higher the mesh level selected, the more cells are generated, and this allows for a more detailed visualisation of the fluid flow. Figure 7 also illustrates the cell density at each mesh level, where the impeller has the highest cell density.

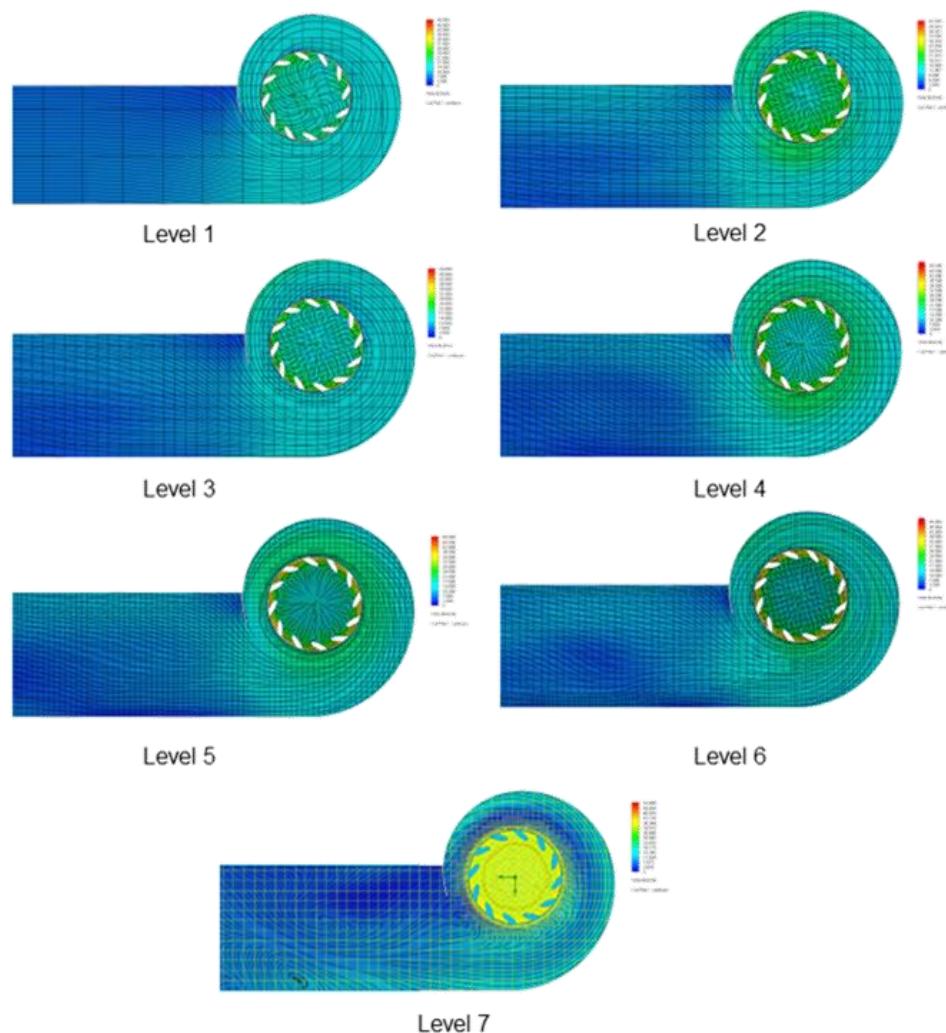


Figure 7. Flow conditions at mesh level settings, a) Level 1, b) Level 2, c) Level 3, d) Level 4, e) Level 5, e) level 6, and f) Level 7

Through the visualisation of the fluid flow, we can understand that the mesh with smaller cells tends to show less accurate flow results, especially for the airflow occurring at the impeller. This can be seen from the lower flow rate entering the impeller compared to the higher mesh levels. However, there is a discrepancy at mesh level 7, where the flow rate increases significantly over the previous six levels. This is due to the cell count being too high, resulting in errors in the calculations. This is also the cause of the difference in the observed results. Therefore, the selection of a very high mesh level may not be appropriate to obtain more accurate results. Proper selection can be achieved through the mesh independency test in this study.

For CFD simulation settings for centrifugal fans in SolidWorks Flow Simulation, it is recommended to use the parameters presented in Table 3.

Table 4. Recommended Solidworks Flow simulation parameter setup

Analysis Type	Fluid		Wall Condition		Turbulence parameter		
	Project Fluid	Flow Type	Thermal Condition	Roughness	Intensity	length	
Internal Rotation Local Region (Sliding)	Air	Laminer & Turbulent	Adiabatic	0 μin	2%	0.01748 mm	
Input Data							
Computational Domain							
Type	Forward	Aft	Span	Upper	Lower	Level	Cell
3D	Auto	Auto	Auto	Auto	Auto	5	214569,00
Calculation Control Setup							
Criterion to Stop	Goal Convergence		Psyhcla Time		Refinement		
All Satisfied	All Goals		1 S		Disable		

4. Conclusion

In this study, CFD simulations with varying mesh levels were conducted to find the optimal mesh type and number. The simulation results are then compared with experimental results to validate their accuracy. The results show that Solidworks Flow Simulation is capable of predicting fluid flow in centrifugal fans and can be considered as an alternative CFD simulation tool that is worth using. Of the various mesh levels tested, mesh level 5 is recommended. Although there is a difference of 16% in total pressure and 0.125% in efficiency between CFD simulation results and experiments, the simulation results at various mesh levels also indicate that a higher number of cells in the mesh does not always produce more accurate results.

Acknowledgements

The authors thank Mechanical Engineering lecturer and Laboratory of Basic Machine Phenomena and Laboratory of Manufacturing Universitas Negeri Padang for its assistance during the simulate and preparation of this manuscript

Declarations

Author contribution

Salmat: Performed the simulation and wrote the original article; Delima Yanti Sari: Formulated a research concept and editing; Yolli Fernanda: Analysis and interpretation of data and Review; Febri Prasetya: Analysis and interpretation of data.

Funding statement

This research received no specific grant from any funding agency in the public, commercial, or not-for-profit sectors.

Conflict of interest

This author declares that they have no competing interests.

Ethical Clearance

There are no human subjects in this manuscript and informed consent is not applicable.

References

- Cory, W. B. (2005). Fans and Ventilation: A Practical Guide. *Fans and Ventilation: A Practical Guide*, 1–424. <https://doi.org/10.1016/B978-0-08-044626-4.X5000-1>
- Eck, B. (1952). Ventilatoren. In *Ventilatoren*. Springer Berlin Heidelberg. <https://doi.org/10.1007/978-3-642-52714-2>
- Huang, C. K., & Hsieh, M. E. (2009). Performance analysis and optimized design of Backward-Curved airfoil centrifugal blowers. *HVAC and R Research*, 15(3), 461–488. <https://doi.org/10.1080/10789669.2009.10390846>
- Kulkarni, S., Chapman, C., & Shah, H. (2016). Computational fluid dynamics (CFD) mesh independency study of A straight blade horizontal Axis tidal turbine. <https://www.preprints.org/manuscript/201608.0008>
- Lam, C. K. G., & Bremhorst, K. (1981). A Modified Form of the k- ϵ Model for Predicting Wall Turbulence. *Journal of Fluids Engineering*, 103(3), 456–460. <https://doi.org/10.1115/1.3240815>
- Lee, Y. T., Ahuja, V., Hosangadi, A., Slipper, M. E., Mulvihill, L. P., Birkbeck, R., & Coleman, R. M. (2011). Impeller design of a centrifugal fan with blade optimization. *International Journal of Rotating Machinery*, 2011. <https://doi.org/10.1155/2011/537824>
- Lin, S. C., & Huang, C. L. (2002). An integrated experimental and numerical study of forward-curved centrifugal fan. *Experimental Thermal and Fluid Science*, 26(5), 421–434. [https://doi.org/10.1016/S0894-1777\(02\)00112-7](https://doi.org/10.1016/S0894-1777(02)00112-7)
- Meng, F., Dong, Q., Wang, Y., Wang, P., & Zhang, C. (2013). Numerical Optimization of Impeller for Backward-Curved Centrifugal Fan by Response Surface Methodology (RSM). *Research Journal of Applied Sciences, Engineering and Technology*, 6(13), 2436–2442. <https://doi.org/10.19026/RJASSET.6.3719>
- Meng, F.-N., Wang, L.-W., Xie, G.-Z., Zhaou, F., Zhang, D.-H., & Du, W.-L. (2017). Effects of Blade Inlet Angle on Flow Field of Centrifugal Fan. *DEStech Transactions on Engineering and Technology Research*, 0(icmeca). <https://doi.org/10.12783/DTETR/ICMECA2017/11924>
- Nabawi, R. A., Syahril, & Primawati. (2021). Study Reduction of Resistance on The Flat Hull Ship of The Semi-Trimaran Model: Hull Vane Vs Stern Foil. *CFD Letters*, 13(12), 32–44. <https://doi.org/10.37934/cfdl.13.12.3244>

- Oberkampf, W. L., & Trucano, T. G. (2002). Verification and validation in computational fluid dynamics. *Progress in Aerospace Sciences*, 38(3), 209–272. [https://doi.org/10.1016/S0376-0421\(02\)00005-2](https://doi.org/10.1016/S0376-0421(02)00005-2)
- Odyas, P., & Moczko, P. (2022). *The New Method of Regulation of Centrifugal Fan*. <https://doi.org/10.26083/TUPRINTS-00021687>
- Patel, J. I., Makadia Engineering, R. N., & Student, P. G. (2014). Review on Performance of Industrial Induced Draft Centrifugal Fan Using Ansys Cfx. *Paripex-Indian Journal of Research*, 1. <https://doi.org/http://dx.doi.org/10.15373/22501991/JAN2014/21>
- Seo, S. J., & Kim, K. Y. (2009). Design Optimization of Forward-Curved Blades Centrifugal Fan With Response Surface Method. *Proceedings of the ASME Heat Transfer/Fluids Engineering Summer Conference 2004, HT/FED 2004*, 2 A, 551–556. <https://doi.org/10.1115/HT-FED2004-56261>
- Singh, O. P., Khilwani, R., Sreenivasulu, T., & Kannan, M. (2011). Parametric Study Of Centrifugal Fan Performance: Experiments and Numerical Simulation. In *International Journal of Advances in Engineering & Technology* (Vol. 33, Issue 2). https://doi.org/10.7323/ijaet/v1_iss2
- Sobachkin, A. (2014). Numerical Basis of CAD-Embedded CFD. https://www.solidworks.com/sw/docs/flow_basis_of_cad_embedded_cfd_whitepaper.pdf
- Thakur, S., Lin, W., & Wright, J. (2002). Prediction of Flow in Centrifugal Blower Using Quasi-Steady RotorStator Models. *Journal of Engineering Mechanics*, 128(10), 1039–1049. [https://doi.org/10.1061/\(ASCE\)0733-9399\(2002\)128:10\(1039\)](https://doi.org/10.1061/(ASCE)0733-9399(2002)128:10(1039))
- Wallace, J. S. (2019). Investigation of SOLIDWORKS Flow Simulation as a Valid Tool for Analyzing Airfoil Performance Characteristics in Low Reynolds Number Flows. *ShareOk*. <https://shareok.org/handle/11244/321619>
- Yu, Z., Li, S., He, W., Wang, W., Huang, D., & Zhu, Z. (2005). Numerical simulation of flow field for a whole centrifugal fan and analysis of the effects of blade inlet angle and impeller gap. *HVAC and R Research*, 11(2), 263–283. <https://doi.org/10.1080/10789669.2005.10391137>